REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave Blank)*	2. REPORT DATE *(MM/DD/YYYY)*	3. REPORT TYPE AND DATES COVERED
	02/01/00	Final Technical Report, 09/29/95 – 09/30/99

4. TITLE AND SUBTITLE
Basic Research for Coupled Multidisciplinary Computations

5. FUNDING NUMBERS
DAAH04-95-C-0067

6. AUTHOR(S)
S.V. Ramakrishnan and K.H. Lee

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESSES
Rockwell Science Center, LLC
1049 Camino Dos Rios
Thousand Oaks, CA 91360

8. PERFORMING ORGANIZATION REPORT NUMBER
SC71116.RFRFTV

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U.S. Army Research Office
P>O> Box 12211
Research Triangle Park, NC 27709-2211
Attn: Patsy S. Ashe, Contracting Officer

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
ARO 34781.1-EG

11. SUPPLEMENTARY NOTES Meaningful Color Pages
The views, opinions and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy or decision, unless so designated by other documentation.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; Distribution Unlimited

12b. DISTRIBUTION CODE
UL

13. ABSTRACT *(Maximum 200 Words)*
A methodology developed for multidisciplinary simulation of the trajectory of a propelled projectile maneuvered by deflecting the canards is presented. An approach to include the effect of the interaction between aerodynamics and structural dynamics is demonstrated. The *quasi-unsteady* approach, briefly discussed here, has the potential to reduce the turnaround time by a factor of around 5 and thus substantially reduce the cost of simulating the trajectory of a projectile including the interaction between fluid dynamics, rigid-body dynamics and structural dynamics.

14. SUBJECT TERMS
CFD, Multi-disciplinary simulation

15. NUMBER OF PAGES
7

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)

20000707 061

Rockwell Science Center

BASIC RESEARCH FOR COUPLED MULTIDISCIPLINARY COMPUTATIONS

Abstract

A methodology developed for multidisciplinary simulation of the trajectory of a propelled projectile maneuvered by deflecting the canards is presented. An approach to include the effect of the interaction between aerodynamics and structural dynamics is demonstrated. The *quasi-unsteady* approach, briefly discussed here, has the potential to reduce the turnaround time by a factor of around 5 and thus substantially reduce the cost of simulating the trajectory of a projectile including the interaction between fluid dynamics, rigid-body dynamics and structural dynamics.

Introduction

This document summarizes the work performed under contract DAAH04-95-C-0067, from the U.S. Army Research Office. Details of the methodologies developed together with extensive analysis of the results obtained have been communicated periodically to Drs. Peter Plostins and Jubaraj Sahu of ARL. This document presents a brief review of the problem studied and the results obtained. A plan for the continuation of this contract is also presented.

The main objective of this contract was to study and develop methodologies to enable real-time multidisciplinary simulation of the trajectory of a projectile. Towards this end the following four tasks were performed during the duration of the contract, 1995-1999.

Task1, the Quasi-Unsteady approach:

Simulation of the trajectory of a projectile involves coupling of fluid dynamics and rigid-body dynamics. Fluid dynamics determines the forces and moments that act on the projectile due to its motion through air, while rigid-body dynamics determines the projectile's response to such forces and moments. These two branches of physics, namely, fluid dynamics and rigid-body dynamics, are tightly coupled in the case of the motion of a projectile. While the relative motion of the projectile with respect to the surrounding air determines the fluid-dynamic forces and moments that act on it, the eventual motion of the projectile relative to air itself is determined by the fluid-dynamic and gravitational, forces and moments.

One of the issues involved in the coupling of fluid dynamics and rigid-body dynamics is the fact that the temporal resolution required for fluid dynamic computations is about 2 to 3 orders of magnitude larger than that required for rigid-body dynamics. In a typical situation, fluid dynamics simulation may require a temporal resolution of 1 microsecond, while 1 millisecond may be sufficient for rigid-body dynamics. This implies that a typical simulation of the trajectory of a projectile with a flight time of 100 milliseconds would require around 100,000 fluid-dynamic time-steps. *Quasi-unsteady* approach has the potential to reduce this by a factor of around 5 without adversely affecting the accuracy of the simulation.

Quasi-steady approach, which is predominantly employed in wind-tunnel tests, can only account for the translational velocities of a projectile while measuring the aerodynamic loads. Both translational and rotational velocities are employed in determining the new location and attitude of the projectile. It is indeed very difficult to account for the rotational velocities in a wind-tunnel test. On the other hand, in the case of a numerical simulation it is quite simple, since only a small change to the boundary conditions employed is required. The *quasi-unsteady* approach, studied under this contract, accounts for both rotational and translational velocities while computing the aerodynamic forces and moments. This approach increases the accuracy of the simulation with negligible increase in computational efforts.

The USA code, a software that solves numerically the Reynolds-Averaged Navier-Stokes equations on structured-grids, was employed in this study. The USA code employs a finite-volume based upwind TVD (Total Variation Diminishing) scheme to generate a numerical approximation for the set of governing partial differential equations. The details of this solver may be found in reference 1. Effects of turbulence are simulated in the USA code using a pointwise one-equation turbulence model (Ref. 2). Additional boundary conditions required to employ the *quasi-unsteady* approach were implemented in the USA code. A six-degree-of-freedom solver developed earlier under a contract from NAWC, China Lake, was used to solve the equations that govern the motion of a rigid body. An interface to couple the six-degree-of-freedom solver with the USA code was developed under this contract.

The trajectory of a high-speed finned projectile with a flight Mach number of around 4.5 was simulated using the quasi-unsteady approach. The results were compared with available range data.

<u>Task2, methodology for numerical simulation of the trajectory of a projectile maneuvered by deflecting forward horizontal fins</u>:

The situation considered here is one wherein the forward horizontal fins are deflected nose-up by a relatively small angle, say 5^0, over a small interval of time, say 30 milliseconds, and then held in that attitude for the rest of the flight. If we were to perform a time-accurate numerical simulation for this case, we would have to rotate the horizontal fins after every time step to their new attitude relative to the projectile and modify the grid to account for the fins' rotation. An approximate method that eliminates the need for the time-consuming regridding without reducing the accuracy of the simulation is to employ a technique similar to the one employed in thin airfoil theory. In cases where the net movement of the fin is small compared to the size of the projectile, it is possible to apply the correct boundary conditions on the original geometry of the fins, thus eliminating the need for expensive regridding. Such a technique was developed under this task, and computations were carried out to demonstrate the application of this technique.

Task3, methodology for numerical simulation of the trajectory of a propelled projectile maneuvered by deflecting forward horizontal fins:

This task is similar to Task2 except that in this case the projectile has its own propulsion. This implies that the numerical simulation has to include real-gas effects with multiple species. The USA code has an extensively tested real-gas model that has the ability to simulate flows with chemical reactions. For this particular task the effect of chemical reactions were neglected considering the fact that the extent of chemical reaction that takes place in an exhaust plume is quite small. The exhaust plume was modeled as a single species with properties obtained from mass averaging the properties of the constituent species. The flow field thus consisted of two species, namely, air and the exhaust gas. The main features of this simulation are correction to the perfect-gas equation of state, non-constant nature of the ratio of specific heats, and additional conservation equation for the exhaust gas.

Task4, aero-structure interaction studies:

The main objective of this task is to compute flow past a projectile, taking into account the interaction between the aerodynamic forces and the structural response of the projectile. Due to funding limitations, only a demonstrative simulation to illustrate the interaction between aerodynamic loads and structural deflections was carried out.

The approach employed here is to compute aerodynamic forces using the USA code, employ *the generalized modal* approach to determine the structural response due to the aerodynamic forces, reevaluate aerodynamic forces taking in to consideration computed change in the shape of the projectile due to the structural response, and then repeat the steps.

Generalized modal approach is a standard, well-known technique employed in the dynamic analysis of structures. It is derived from linearizing the governing equations. Any standard structural analysis software, such as ABAQUS, may be employed to obtain the dynamic response of a structure to applied external forces.

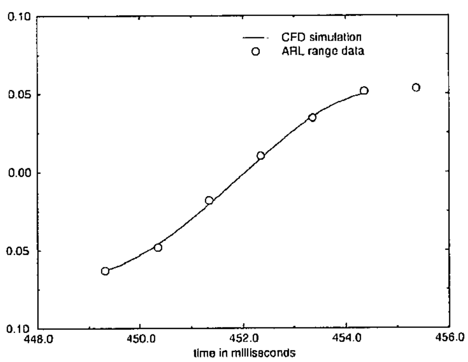
Summary of results

Task1, the Quasi-Unsteady approach:

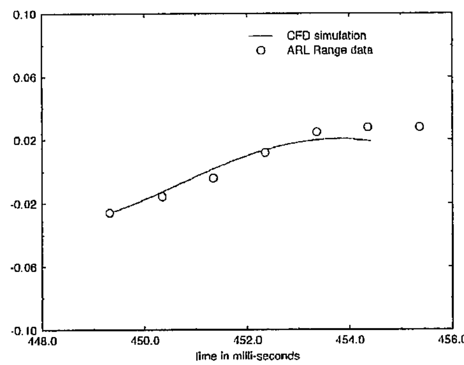
Under this task, boundary conditions required to employ the *quasi-unsteady* approach were developed and implemented in the USA code, a structured-grid CFD

(Computational Fluid Dynamics) solver. An interface to couple the USA code and a rigid-body dynamics solver was also developed. Simulations were carried out for a high-speed finned projectile (Mach No. ≈ 4.5) for which trajectory data was available. The geometry of the projectile is shown in Fig. 1. Results from the numerical simulation were compared with measurements (Fig. 2) and it was shown that the quasi-unsteady approach is indeed suitable for such simulations. It was also demonstrated that the use of quasi-unsteady approach reduced the turnaround time for trajectory simulation by a factor of around 5.

Fig. 1. Projectile geometry



Side-slip vs. time



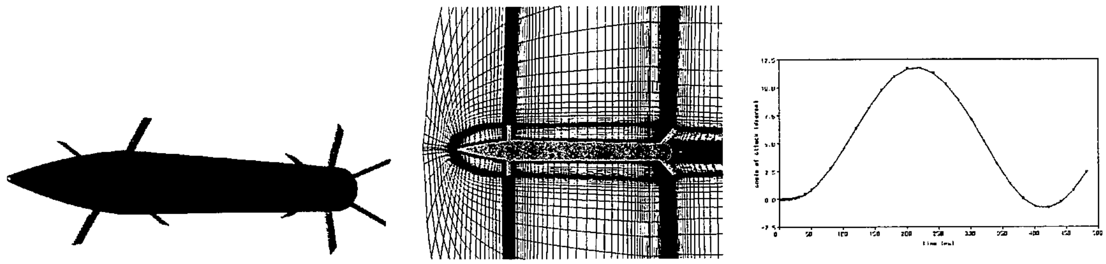
Angle-of-attack vs. time

Fig. 2. Validation of *quasi-unsteady approach* for a high-speed projectile

<u>Task2, methodology for numerical simulation of the trajectory of a projectile maneuvered by deflecting forward horizontal fins</u>:

An extended range projectile with fore and aft fins was employed in the demonstration of the use of quasi-unsteady approach in the simulation of the trajectory of a projectile maneuvered by deflecting forward horizontal fins (canards). The canards were deflected nose-up by 5^0 during the first 30 milliseconds of the trajectory and then held fixed at that attitude for the reminder of the simulation. Initial conditions for the trajectory simulation correspond to steady horizontal flight at an altitude of 8 kilometers, and Mach number of 1.5.

The geometry of the projectile, the grid employed in the simulation, and the temporal history of the projectile's angle-of-attack are shown in fig.3. No data is available for this geometry to enable validation of the results obtained from the numerical simulations.



(a) (b) (c)

Fig. 3. Numerical simulation of the trajectory of a projectile maneuvered by deflecting forward horizontal fins; a) geometry of the projectile, b) the grid employed in the simulation. and c) temporal history of the projectile's angle of attack.

<u>Task3, methodology for numerical simulation of the trajectory of a propelled projectile maneuvered by deflecting forward horizontal fins</u>:

The geometry employed in this simulation is the same as the one used for Task2. The properties of the exhaust plume were supplied by Dr. Sahu at ARL. The effect of the plume on the normal and axial forces, and the pitching moment are depicted in fig.3. The axial force becomes negative (thrust) when the plume is turned on. The normal force increases slightly due to the contribution from the thrust. There is only a small change in the pitching moment due to the fact that most of the contribution to the pitching moment comes from the deflection of the forward horizontal fins.

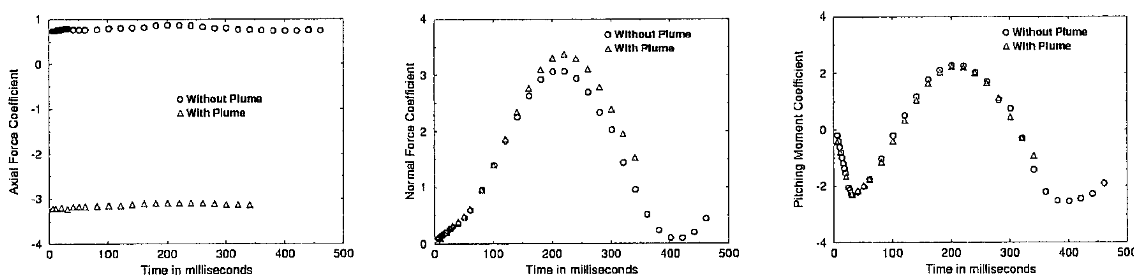


Fig.4. Temporal variation of axial and normal forces and pitching moment

Snap shots of the surface pressure distribution and Mach number contours for the plane of symmetry at four time levels are shown in the following figure. The interaction between the exhaust plume and external flow may be clearly seen from these snapshots.

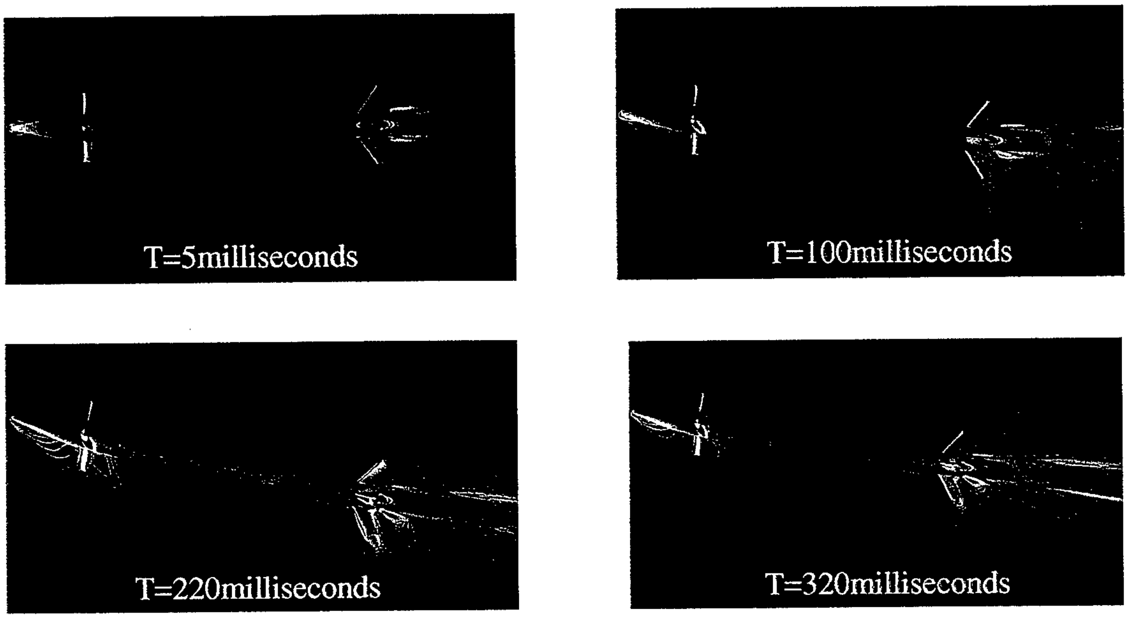


Fig. 5. Snapshots of the surface pressure distribution and Mach number contours on the symmetry plane.

Task4, aero-structure interaction studies:

Only the static interaction between the aerodynamic forces and the structural response was considered under this task. The dynamic response will be the topic of the follow-on contract under negotiation. The commercially available tool for structural analysis, the ABAQUS, was employed in the study. Interfaces to exchange information between the CFD solver, the USA code, and ABAQUS were developed. Only the fin was considered to be structurally flexible. The results of the analysis are shown in the following figures.

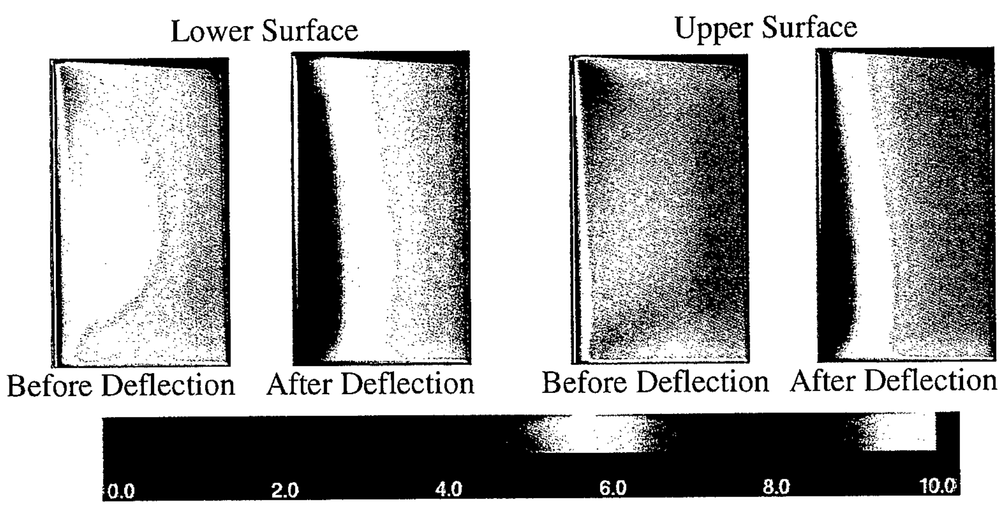


Fig.6. Static Aeroelastic Analysis of the Canard; Normal Stress in pounds/sq.inch

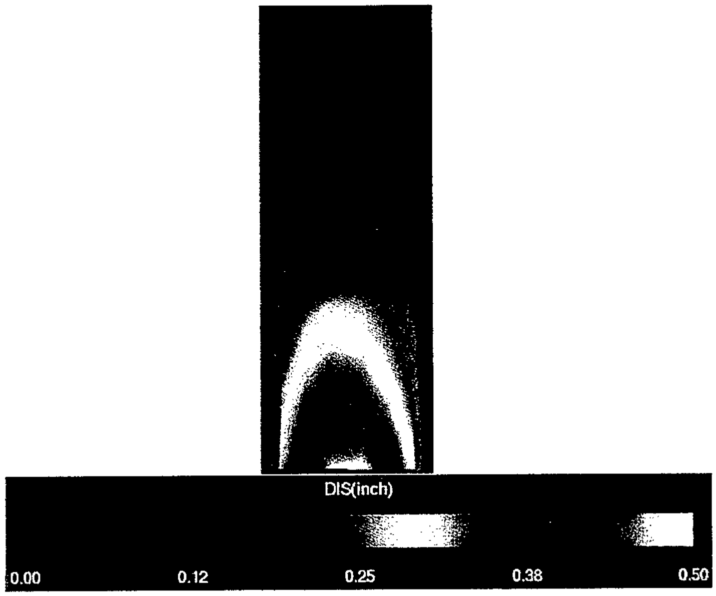


Fig. 7. Static Aeroelastic Analysis of the Canard; Vertical displacement in inches

The last two figures show that the aero-structure interaction increases the normal stress on the forward horizontal fins (the canards) and that the cantilever effect results in maximum deflection at the tip.

Proposed extensions:

1. Perform dynamic analysis of the interaction between aerodynamic forces and structural response for the propelled extended range projectile maneuvered by deflecting the forward horizontal fins.

2. Develop interfaces to automate the interaction between CFD simulation and structural analysis and reduce the turnaround time for trajectory simulation by at least a factor of 2.

References

1. S. R. Charavathy, et. al., "Unified Nose-to-Tail computational method for Hypersonic Vehicle Applications," AIAA paper 88-2564, 1988.

2. U. C. Goldberg and S. V. Ramakrishnan, "A Pointwise Version of Baldwin-Barth Turbulence Model," International Journal of Computational Fluid Dynamics, Vol. 1, No. 4, 321-338.

List of all participating scientific personnel

Dr. S. V. Ramakrishnan, Manager, CFD, Rockwell Science Center.

Mr. K. H. Lee, Member, Technical Staff, CFD, Rockwell Science Center.

Dr. U. C. Goldberg, formerly Member, Technical Staff, Rockwell Science Center, currently, Senior Scientist, Metacomp Technologies.